

Citation for published version:

Benzeval, I, Turner, IG & Ellis, MJ 2013, 'CFD-aided design of a fluidised bed bioreactor for bone tissue engineering', Tissue and Cell Engineering Society Annual Conference (TCES), Cardiff, UK United Kingdom, 23/07/13 - 25/07/13.

Publication date:

2013

Document Version

Early version, also known as pre-print

[Link to publication](#)

University of Bath

Alternative formats

If you require this document in an alternative format, please contact:
openaccess@bath.ac.uk

General rights

Copyright and moral rights for the publications made accessible in the public portal are retained by the authors and/or other copyright owners and it is a condition of accessing publications that users recognise and abide by the legal requirements associated with these rights.

Take down policy

If you believe that this document breaches copyright please contact us providing details, and we will remove access to the work immediately and investigate your claim.

1. Introduction

The aim of this project was to perform Computational Fluid Dynamic (CFD) analysis of liquid-solid fluidisation of relatively large particles to inform the design and development of a fluidised bed bioreactor for the culture and growth of bone cells seeded into porous bioceramic particles. The need for artificial bone graft material, particularly for large defects which cannot be filled with autologous bone, is driving research into synthetic alternatives^{1,2}. The project utilises calcium phosphate particles (4 – 10 mm diameter), with large (500 - 1500 mm diameter) interconnected pores. The fluidisation of the particles was modelled to inform the sizing of the bioreactor and to find a balance between quality of fluidisation and the volume of media required to be pumped.

3a. Results

The CFD simulations are shown in Figures 1 and 2. In Figure 1 the depth of the bed from the simulations are plotted along with the theoretical bed depth at the velocities used. It can be seen that the three laminar simulations (Figure 1a, 1c and 1e) produced stable bed depths at each flow rate similar to the theoretical depth, with the exception that each underwent an unexplained reduction in bed depth at approximately 10, 26 and 15 s respectively. The turbulence simulations did not have this reduction in bed depth. However the two smallest columns (Figure 3b and 3d) included a surge in the bed depth when increased to the highest flowrate (at t = 50 s). Figure 2 shows this slugging behaviour in greater detail. The images are axial cross-sections through the column showing the volume fraction.

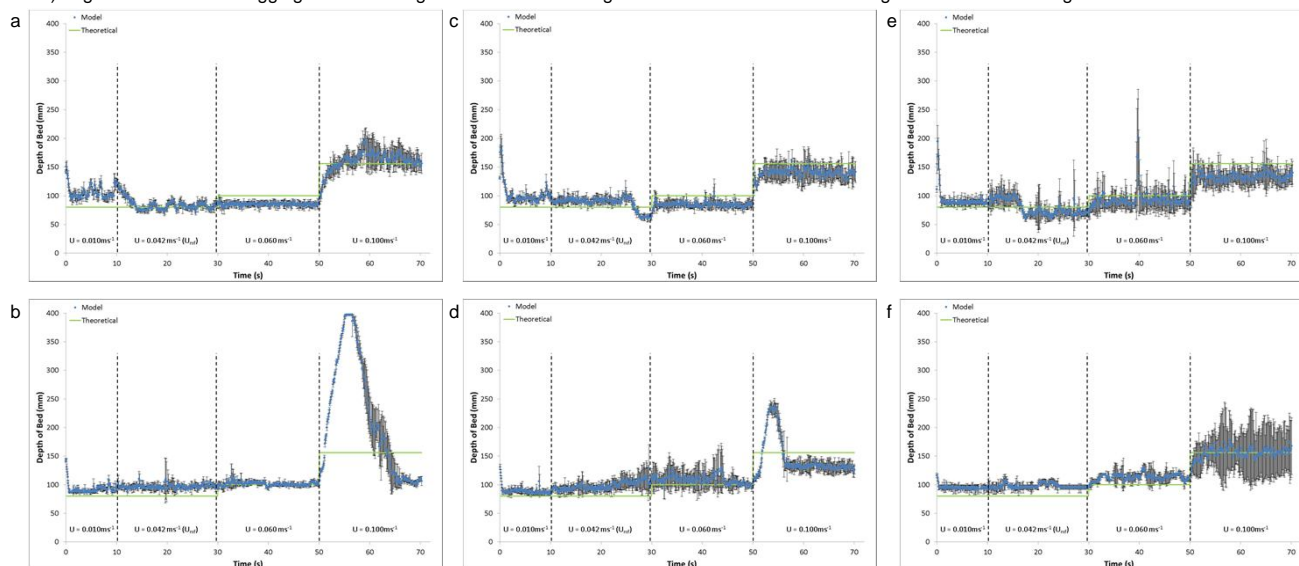


Figure 1: Variation of the depth of bed with time for the six simulations. a) 25 mm diameter, laminar model; b) 25 mm diameter, turbulence model; c) 35 mm diameter, laminar model; d) 35 mm diameter, turbulence model; e) 80 mm diameter, laminar model; f) 80 mm diameter, turbulence model (Error bars ± 1 standard deviation)

4. Discussion

The DDPM was successfully used to model liquid fluidisation of relatively large particles. This was dependent upon two critical factors:

Mesh Size: It was found that for the model to successfully converge, the mesh nodes had to be larger than the particles being fluidised. In this case, with 4 mm diameter particles, a minimum mesh size of 6 mm was required. A mesh size of 5 mm was attempted, but would not converge. This is contrary to the standard approach of CFD modelling, where the mesh would normally be made significantly smaller to improve the accuracy of the liquid phase simulation.

Courant Number: The Courant number is a guideline for calculating the time step to use during a transient simulation, with a value of 1 being the suggested start point. Setting the Courant number to 1 would result in a time step that, given the appropriate linear flowrate, a portion of liquid would cross one mesh node during one time step. In this work it was found that following this guideline resulted in simulations that would not converge. The maximum Courant number used in this work was 8.3×10^{-3} . This equates to a particle or portion of liquid requiring 120 time steps to cross a single mesh node.

The data also shows that in the narrowest columns it is possible to cause a slugging effect which lifts the particles to the top of the column. This effect was only observed in the simulations that included turbulence. This additional movement on the particles could potentially be detrimental to cells being grown within the porous structure. As such large changes in the liquid velocity should be performed gradually.

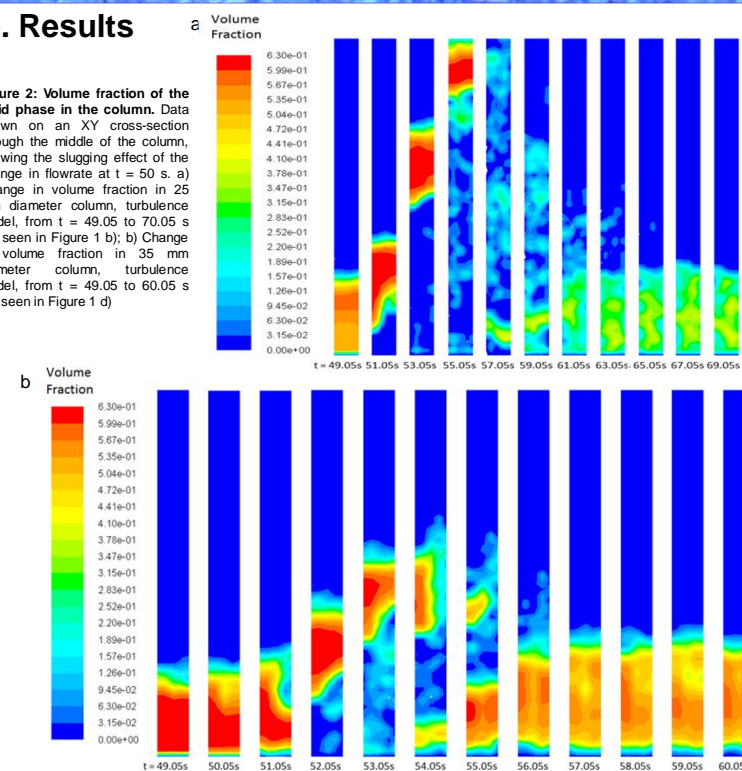
2. Methodology

The CFD simulation of the fluidised bed bioreactor (FBB) was performed in Ansys FLUENT using a dense discrete phase model (DDPM) for the solid phase. The liquid phase was modelled to represent phosphate-buffered saline at 37 °C. The solid phase was modelled as 4 mm glass spheres with a density of 2500 kgm⁻³. The fluidisation was modelled in columns of 25, 35 and 80 mm diameter, all 400 mm high. Each column was simulated twice, initially with a laminar flow model and secondly with a $k-\epsilon$ turbulence model. The flow of liquid was incrementally increased from 0.010 ms⁻¹ to 0.042, 0.060 and 0.100 ms⁻¹ at 10, 30 and 50 s respectively. The theoretical minimum fluidisation velocity of the glass beads, calculated using the Ergun equation, is 0.042 ms⁻¹.

3b. Results

Figure 2: Volume fraction of the solid phase in the column. Data shown on an XY cross-section through the middle of the column, showing the slugging effect of the change in flowrate at t = 50 s. a)

Change in volume fraction in 25 mm diameter column, turbulence model, from t = 49.05 to 70.05 s (as seen in Figure 1 b); b) Change in volume fraction in 35 mm diameter column, turbulence model, from t = 49.05 to 60.05 s (as seen in Figure 1 d)



5. Conclusion

- Liquid-solid fluidisation of relatively large particles has been successfully simulated in 3D using a dense discrete phase model.
- It has been found that the mesh size for simulations of large particles must be greater than these solid phase particles for successful convergence to be possible.
- Guidelines for the size of the time step for this type of transient simulation should be revised so that the Courant number is at most $\sim 10^{-3}$ to ensure convergence.
- Abrupt large changes in the liquid velocity should be avoided to prevent potential slugging effects.

These findings demonstrate the application of CFD modelling to enable the successful design and development of a fluidised bed bioreactor for the culture and growth of bone cells seeded on to porous bioceramic particles.

Refs: ¹ Boyan B. *et al.* 2003. Bone Graft Substitutes: ASTM International ²Stevens MM. 2008. Biomaterials for bone tissue engineering. *Materials Today* 11(5):18-25

Acknowledgements: The authors would like to thank the EPSRC for funding this research (Grant no. EP/I015922/1)

$$Courant\ number = \frac{u\Delta t}{\Delta x}$$

u = linear velocity (ms⁻¹), Δt = time step (s)
 Δx = mesh node length (m)